International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 5, Issue 4, Aug 2015, 33-40

COLD FLOW SIMULATION OF QUENCHING MEDIA IN AGITATED QUENCH TANK WITH DIFFERENT CONFIGURATIONS USING CFD SOFTWARE

DHIRAJ BHIKA CHAUDHARI[1], RAGHUNATH YADAV PATIL[2] & ATUL SHIVAJI CHAUDHARI[3]

[1]Research Scholar, Department of Mechanical Engineering, SGDCOE, North Maharashtra University, Jalgaon, Maharashtra, India

[2]Assistant Professor, Department of Mechanical Engineering, SGDCOE, North Maharashtra University, Jalgaon, Maharashtra, India

[3]Assistant Professor, Department of Mechanical Engineering, Government Polytechnic, North Maharashtra University, Jalgaon, Maharashtra, India

ABSTRACT

In the business of heat treatment quenching process has great role to play. Cooling of the parts at rapid rate but in controlled manner is very essential to obtain the optimum desired mechanical properties. The quenching system with agitation arrangement circulates the quenchant in effective manner and shortens the quenching time.

Many parameters like type of quenching media used, design of quenching tank, bath temperature, agitation system etc. decides the final mechanical properties of work piece. Quench tank design depends on many components of system like draft tube impeller, structural aspects of flow directing baffles and many more. Use of draft tube impellers will results in directional fluid flow around a part surface being quenched The circulation of quenching media in tank depends on many parameters such as use of draft tube , type of impeller used & position of impeller, flow separators in the draft tube

In the present study tank with two agitator system was considered for analysis. Initially the design is kept simple. Simple pipes are used to direct the flow of quenching fluid. Then bend pipe is introduced in the system and the flow patterns & pressure on the job were analyzed. After that initially one flow deflector was used and then three deflectors were used to carry out the analysis. Results of the analysis are in good agreement with the published literature.

KEYWORDS: Cold Flow Simulation, Quench Tank Agitation, Pressure Difference, Optimization

INTRODUCTION

Desired mechanical properties of the many steel & aluminum alloys can be obtained by heat treatment & quenching. In the quenching tank if agitation system is not used then heat transfer takes place due to natural convection. Vaporization of quenching media on the surface of the parts to be quenched occurs & it reduces the heat transfer rate. The agitation system for forced circulation is required to shorten the cooling times. Where control over the cooling rate is important, mechanical agitation provides the best performance at the lowest energy costs [1]. The hardness and depth of hardening during the quench is affected by agitation because of the rupture of the relatively unstable film boiling cooling process that always occur in vaporizable quenchants such as oil, water and aqueous polymers. Hence agitation helps to increase the rate of heat transfer throughout the quenching process regardless of the bath temperature. Also due to agitation production of smaller and more frequent bubbles takes place during boiling stage, which in turn, creates faster cooling

rates. Understanding how quenching parameters affect the outcome of the quench is important for control of mechanical properties as well as elimination of distortion and cracking [3].

Muammer Koc, John Culp, Taylan Altan studied “Prediction of residual stresses in quenched aluminium blocks and their reduction through cold working processes.” In this study, numerical techniques were used to predict residual stresses after quenching of Al 7050 forged block, and the predictions were compared with experimental measurements. N. Lior had observed in “The Cooling Process in Gas Quenching.” that the flow non uniformity in quench chambers is caused primarily by the chamber design hence it can be controlled with proper design of flow passages and CFD modeling and simulation plays important role. Marco Fontecchio, Mohammed Maniruzzaman and Richard D. Sisson, Jr had studied “The Effect of Bath Temperature and Agitation Rate on the Quench Severity of 6061 Aluminium in Distilled Water.” The main objective of this work was to experimentally determine the effect of bath temperature and agitation rate of the quenching medium on cooling behaviour and Quench Factor, Q. Shuhui Ma, Aparna S.Varde, Makkio Takahashi, Darrell.K.Rondeau, Md.Maniruzzaman and R.D.Sisson, Jr. had worked on “Quenching- Understanding, Controlling and Optimizing the Process.” In this work they have described four different quench probe systems and experimental results were presented in terms of cooling rate. D. D. Hall and I. Mudawar published “Predicting the Impact of Quenching on Mechanical Properties of Complex- Shaped Aluminium Alloy Parts.” The aim of the study was to develop an intelligent spray quenching system which selects the optimal nozzle configuration based on part geometry and composition such that the magnitude and uniformity of hardness (or yield strength) is maximized while residual stresses are minimized. N. Bogh had published “Quench Tank Agitation Design Using Flow Modeling” in that guidelines that were used in modeling and measuring an existing quench tank flow with a conventional pumping agitation system has been given. He has represented the method to analyze the quench tank system for modification and given three step processes for the same. It includes equipment inspection, mechanical survey and element analysis. D.R.Garwood, J. D. Lucas, R. A. Wallis and J. Ward have published “Modelling of the Flow Distribution in an Oil Quench Tank”. In this article they have investigated the fluid flow in an agitated quench tank used during heat treatment of superalloy forgings. A commercially available CFD code was employed to predict the flow field within the quenchant. In the experimental investigation they have used four impeller model of tank for agitation purpose. Predictions have been compared with the experimental data obtained on a small –scale water model of the system.

In the present work four different configurations of quench tank system has been taken into account and aim is to find out best configuration which gives optimum quenching properties.

COMPUTATIONAL FLUID DYNAMICS SIMULATION

Governing Equations of Fluid Flow

The basic conservation equations of mass, momentum and energy for incompressible flow problems can be expressed as [8].

Mass Equation: $\frac{\partial \rho}{\partial t} + \text{div}\,(\boldsymbol{\rho} v) = 0$ (1)

Momentum Equation: $\frac{dv}{dt} = \boldsymbol{\rho} F + \frac{\partial Px}{\partial x} + \frac{\partial Py}{\partial y} + \frac{\partial Pz}{\partial z}$ (2)

Energy Equation: $\frac{\partial(\rho T)}{\partial t} + \text{div}\,(\boldsymbol{\rho} vT) = \text{div}\left(\frac{k}{cp}\ grad\ T\right) + S$ (3)

where ρis the fluid density; *t* stands for time; ***v*** is thefluid velocity vector; (Px, Py, Pz) are Cartesian components of the stress tensor ***p***; ***F*** is the body forcevector per unit volume of a fluid particle; *T* is the thermodynamic temperature; *cp* is the specific heat capacity; *k* is the heat transfer coefficient of the fluid; *S*is a source of energy per unit volume per unit time.

Quenching Tank

Outer dimension of the quenching tank considered for the analysis are 2.5 m×2.5 m×3 m. The quenching zone is located at center and having size 1 m×1 m×1 m. The quenching tank consists of two agitators, two impellers, and two draft tubes. Four different configurations of agitation system are considered for analysis. The design is kept simple. Simple pipes are used to direct the flow of quenching fluid. Then bend pipe is introduced in the system and the flow patterns & pressure on the job were analyzed. After that initially one flow deflector was used and then three deflectors were used to carry out the analysis as shown in figures below. Six small cylindrical jobs of A357 alloy work pieces are considered. The impeller used in agitation system has three blades with pitch setting of 65 mm. The outer diameter of impeller is 410 mm.

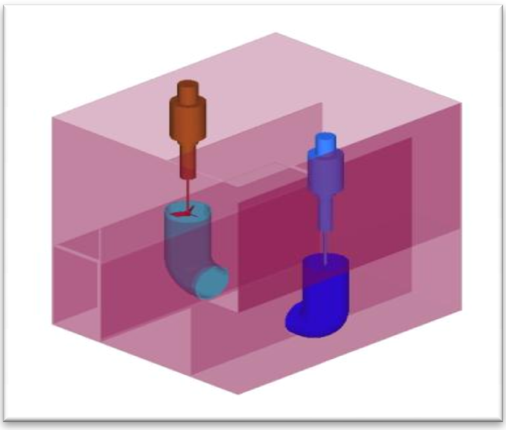
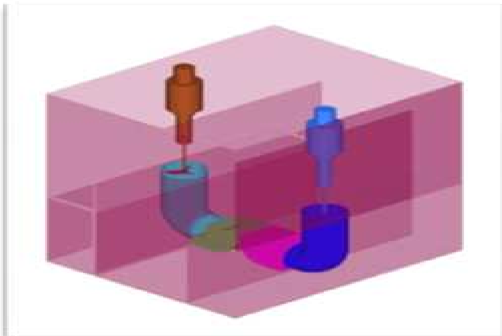
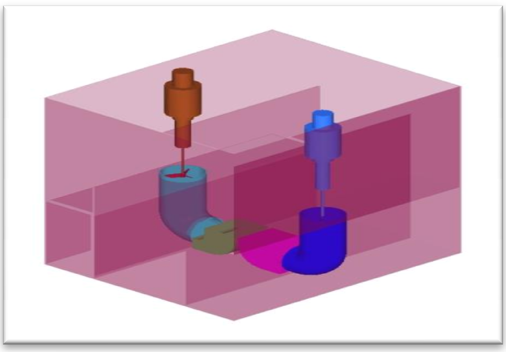
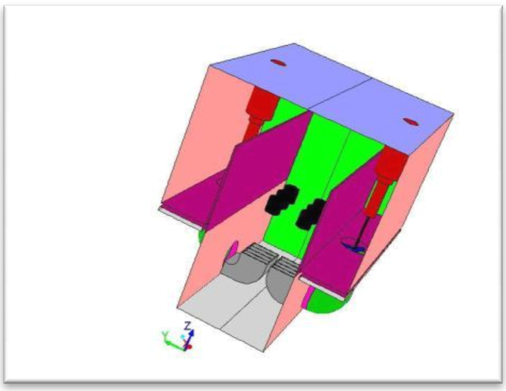
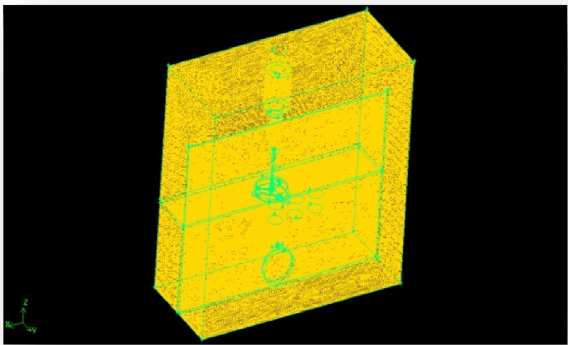
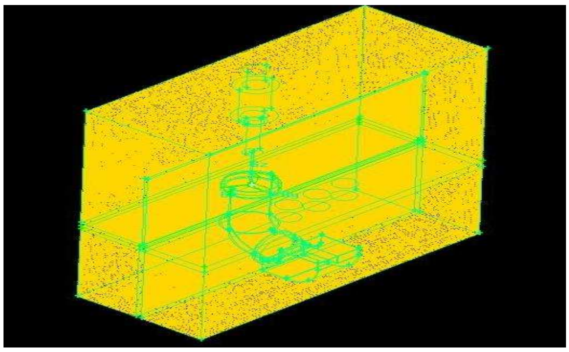
	
Case 1: Quenching Tank with Simple Agitation System	**Case 2: Quenching Tank with Simple Agitation System & Bend Pipe**
	
Case 3: Quenching Tank with Simple Agitation System & Bent Pipe with One Separator in Pipe	**Case 4: Quenching Tank with Simple Agitation system & Bent pipe with Three Separators in Pipe**

Figure 1: Geometric Models of Four Cases under Considerations

Numerical Simulation

GAMBIT software is used for flow zone modeling and meshing. The Fluent14.5 software is used to simulate liquid flow distribution in quenching tank. The Gambit software is important tool to create the CFD models. Tet / Hybrid elements, Tgridtype and interval size of 0.05 were selected for the quenching tank, which can get a good grid quality andthe total nodes are 67926, and the total elements are 365666. Compared with quenching tank, the structure ofdraft tube is a complex and small part, so its mesh grid parameters were selected as follows: Tet / Hybridelements, Tgrid type and interval size of 0.04 (total nodes 3573, total elements 16136)an important tool to create the computational fluiddynamics (CFD) models.Tet/Hybrid elements, Tgridtype and interval size of 0.05 were selected for thequenching tank, which can get a good grid quality andthe total nodes are 67926, and the total elements are365666. Compared with quenching tank, the structure ofdraft tube is a complex and small part, so its mesh gridparameters were selected as follows: Tet/Hybrid elements, Tgrid type and interval size of 0.04 (totalnodes 3573, total elements 16136). Figure 4 shows the mesh grid of simulation model.

Water was selected as the quenching medium. The physical properties of water at 25 °C are as follows: density (ρ=997.04 kg/m3) and viscosity (μ=8.904×10−4Pas). The continuum hypothesis and the nonslip condition at the walls are applicable. In numerical simulation, the forward propulsion force of impeller agitation is only considered and the rotational force is ignored. The coordinate system is shown in Figure 4. x-velocity (v_x) and z-velocity (v_z) of inlet are all zero. y-velocity (v_y) of inlet can be calculated using the impeller parameters, including impeller diameter (d450), number of blades (N=3), pitch setting (p=65 mm) and rotational speed is taken as 1200 rpm . At the outlet, the pressure is set as one standard atmospheric pressure. Viscosity coefficient and density of water are set as constant.

	
Case 1: Quenching Tank with Simple Agitation System	**Case 2: Quenching Tank with Simple Agitation System & Bend Pipe**

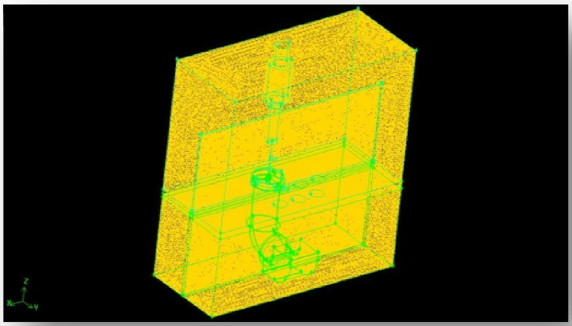
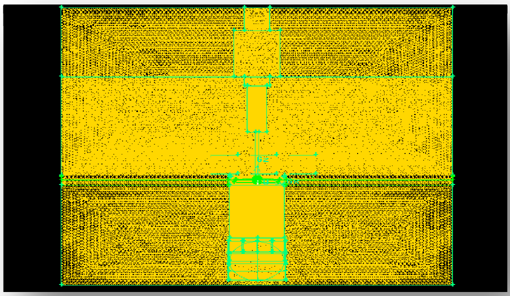
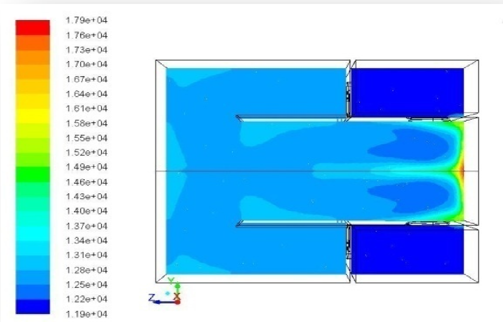
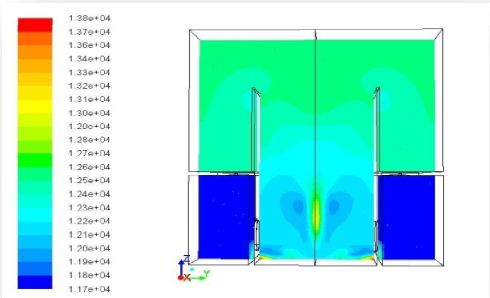
	
Case 3: Quenching tank with Simple Agitation System & Bent Pipe with One Separator in Pipe	**Case 4: Quenching Tank with Simple Agitation System & Bent Pipe with Three Separators in Pipe**

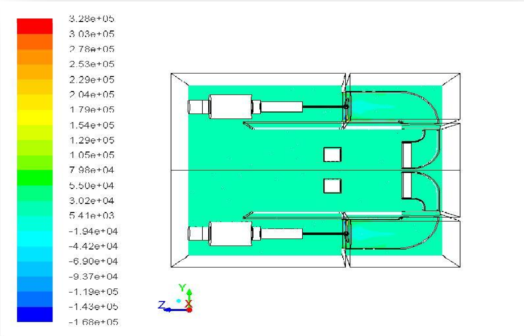
Figure 2: Meshed Model of Four Configurations under Consideration

RESULTS

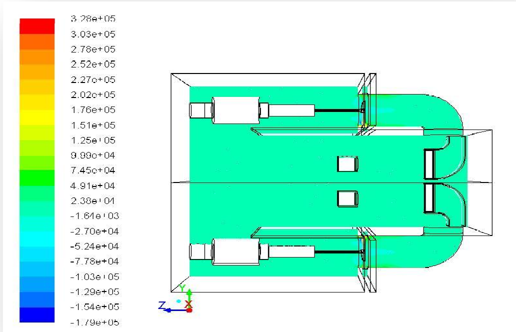
Variation of Total Pressure

The variation of total pressure for all the four cases is as shown in figure. The pressure variation in the case no 4 i.e. agitation system with bent pipe and three flow deflector is more uniform than other cases. & nature of the pressure distribution is in good agreement with the published literature.

	
Case 1: Total Pressure Distribution in Quenching Tank with Simple Agitation System	**Case 2: Total Pressure Distribution in Quenching Tank with Simple Agitation System & Bent Pipe**



Case 3: Total Pressure Distribution in Quenching Tank with Simple Agitation System & Bent Pipe with One Separator in Pipe

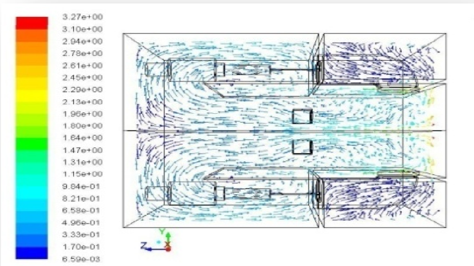


Case 4: Total Pressure Distribution in Quenching Tank with Simple Agitation System & Bent Pipe with Three Separators in Pipe

Figure 3: Total Pressure Distribution in Quenching Tank with Four Cases under Consideration

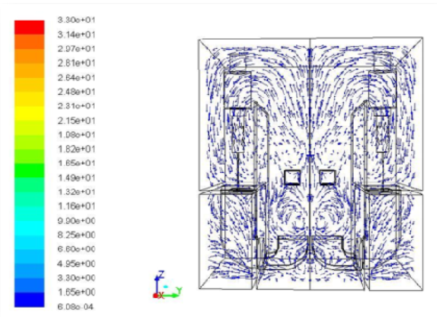
Variation of Velocity

Velocity of quenchant circulation plays important role in the time required to accomplish quenching process. For simulation agitator speed is considered as 1200 RPM. The pattern of velocity vector variation is in good agreement with the published literature. Turbulence in the case no 4 is least in all cases and thus the quenching time will be least in case no 4.

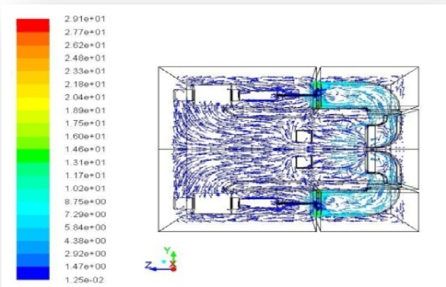


5

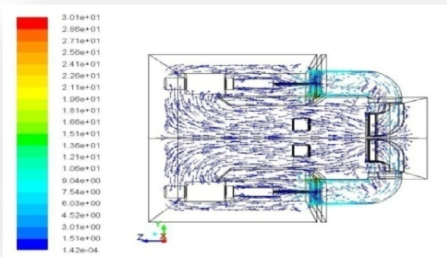
Case 1: Velocity Distribution in Quenching Tank with Simple Agitation System



Case 2: Velocity Distribution in Quenching Tank with Simple Agitation System & Bent Pipe



Case 3:Velocity Distribution Quenching Tank with Simple Agitation System & Bent Pipe with One Separator in Pipe



2.5

Case 4: Velocity Distribution in Quenching Tank with Simple Agitation System & Bent Pipe with Three Separators in Pipe

Figure 4: Velocity Distribution in Quench Tank with Four Cases under Consideration

Static Pressure on Work Piece

The figures below show the static pressure acting on the all surfaces of the job to be quenched. In case no1, 2 & 3 the pressure distribution acting on the faces of the job is not uniform. High pressure points are observed on the tot & bottom faces of the job. These high pressure points results in non uniform quenching of the job & thus will affect the metallurgical properties of the job. In case no 4 the distribution of the pressure is much uniform which will result in uniform quenching.

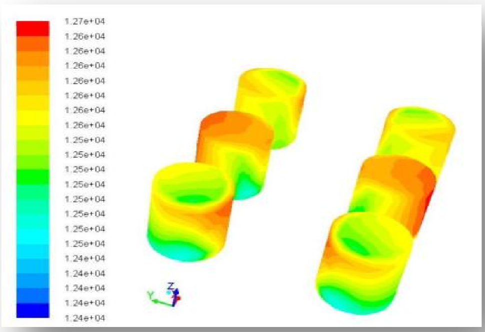
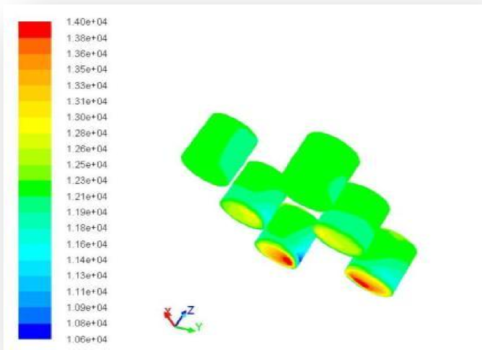
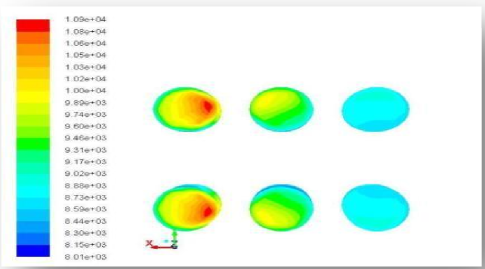
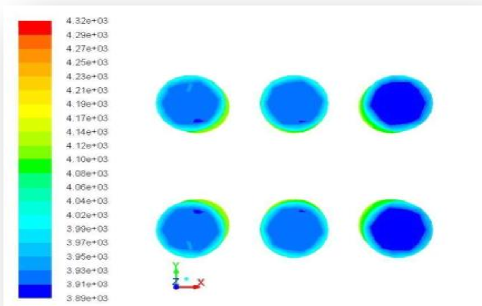
	
Case 1: Static Pressure Distribution Acting on Workpiece with Simple Agitation System	**Case 2: Static Pressure Distribution Acting on Work Piece with Simple Agitation System & Bent Pipe**
	
Case3: Static Pressure Distribution Acting on Workpiece with Simple Agitation System & Bent Pipe with One Separator in Pipe	**Case4: Static Pressure Distribution Acting on Workpiece with Simple Agitation System & Bent Pipe with Three Separators in Pipe**

Figure 5: Total Pressure Distribution Acting on Workpiece in Four Cases under Consideration

DISCUSSION & CONCLUSIONS

CFD analysis for different cases of agitation system was carried out & it predicts the values of total pressure acting on the surface of the component, velocity components of the quenching fluid with reasonably good accuracy. The CFD results for different agitation system provides quick and cost effective alternative for experimental study. Following conclusions can be drawn from the above study.

- Nature of pressure distribution counters & velocities in the domain are in good agreement with the published literature.
- Addition of pipe, bent pipe & deflectors in the bend pipe in the agitation system were analyzed. These

modifications directly affect the flow of quenching media.

- In first there cases spots of higher pressure can be seen where as pressure distribution over the job is uniform in fourth case
- The performance of the case no 4 i.e. agitation system with bent pipe having three flow separators is best among all four cases with less variation of pressure across top, Side and bottom face of the Work piece and good velocity distribution in quenching zone.

REFERENCES

1. S.M. Adedayo et.al. (2014) "Effect of Quench Immersion Speed in Water on the Mechanical Properties of C30 Carbon Steel." Proceedings of the World Congress on Engineering 2014 Vol. II, WCE 2014, July 2 - 4, 2014, London, U.K.
2. Muammer Koc et.al. (2006) "Prediction of residual stresses in quenched aluminum blocks and their reduction through cold working processes." Journal of Materials Processing Technology 174 , 342–354
3. Marco Fontecchio et.al. (2003) "The Effect of Bath Temperature and Agitation Rate on the Quench Severity of 6061 Aluminum in Distilled Water." Proceedings of the 1st ASM International Surface Engineering and the 13th IFHTSE Congress (ASM International) 449 - 456 (8).
4. N.Lior (2004), "The Cooling Process in Gas Quenching." Journal of Materials Processing Technology 155–156, 1881–1888.
5. Shuhui Ma et.al. (2003), "Quenching- Understanding, Controlling and Optimizing the Process." Proceedings of the Fourth International Conference on Quenching and the Control of Distortion, 20-23 May, 2003, Beijing
6. D. D. Hall and I. Mudawar (1995), "Predicting the Impact of Quenching on Mechanical Properties of Complex-Shaped Aluminium Alloy Parts." Journal of Heat Transfer, Vol. 117 / 479.
7. BOGH N. Quench tank agitation design using flow modeling. Heat Treating: Equipment and Processes: Conference Proceedings Ohio: ASM International, 1994: 82−91.
8. Xia-wei Yang et.al. (2013), "Optimum design of flow distribution in quenching tank for heat treatment of A357 aluminum alloy large complicated thin-wall workpieces by CFD simulation and ANN approach."